Chapter 3

Simulation of Non-Ideal Components in LTSpice

3.1 Pre-Lab

The answers to the following questions are due at the beginning of the lab. If they are not done at the beginning of the lab, no points will be awarded.

1. How do you expect the frequency generator output voltage to change with a ”heavier” load (decreasing impedance)? Why?
2. Explain the meaning of component tolerance. (E.g. Tolerance of a resistor)
3. Is a real inductor completely reactive? Why or why not?

3.2 Section Overview

The engineering method always begins with an initial design that is implemented and tested. Once the design is complete and before the circuit is ever implemented, the engineer will simulate its behavior using a tool such as LTSpice. LTSpice is a great tool for simulating circuits so that you can know how they will respond before you build them. However, simulating ideal circuit components is not the complete picture. Circuits in the real world are not ideal; they contain parasitic components. In this lab the non-ideal characteristics of components will be explored. LTSpice will be used to account for these slight differences between real and ideal parts.

3.3 Non-Ideal Resistors

Reality is messy. Most lecture material covers electronics from an ideal point of view. Only in lab when using real components does it matter that parts have tolerances. Even resistors have an accuracy tolerance that can vary their value by as much as 10%. To demonstrate this property, start by calculating V_{out} in the ideal circuit in figure 3.1 and enter it below.

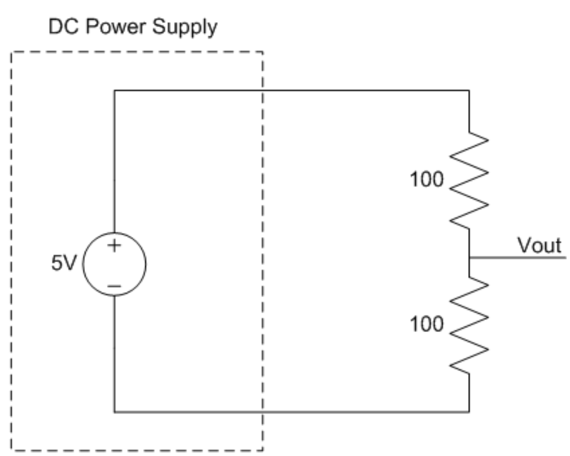

Figure 3.1: Voltage Divider

What is the expected (calculated) voltage at V_{out}? ______________________(1)

All resistors have an associated tolerance. The through-hole resistors included in your kits have colored bands to represent their value as well their tolerance. Look at the last color band on the resistor to determine the accuracy tolerance according to table 3.1.

4^{th} Band Color	Accuracy Tolerance
Brown	1%
Red	2%
Orange	3%
Yellow	4%
Gold	5%
Silver	10%

Table 3.1: Resistor Tolerance

This percentage is representative of the percent resistance that the resistor might be off from its stated value. For instance a 1kΩ resistor with a tolerance of 3% could possibly be $(1000\Omega)(.03) = 30\Omega$ off from 1kΩ. As you can imagine this could change your expected results by a substantial amount.

Calculate the possible difference in resistance due to tolerance of the resistors used in the circuit:
__________________________(2)

Using an ohmmeter, measure the actual values of the resistors and record them on the circuit in figure 3.2. Now construct the circuit from figure 3.2. Measure the voltage being supplied by the power supply and record it on the circuit. Finally, measure the voltage at V_{out} and record it as well.

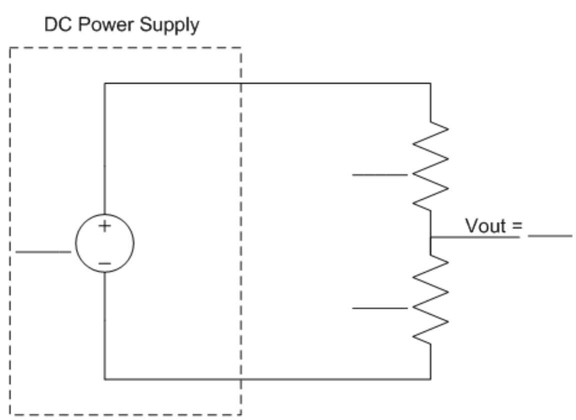

Figure 3.2: Fill in Voltage Divider

Compare the two circuits (ideal and non-ideal) and see how there are slight differences. In most cases these small differences don't have a big effect on a circuit, but in many cases (e.g. AC sources at very high frequencies) the non-ideal properties of components have to be considered.

3.4 Non-Ideal Voltage Source

An ideal voltage source is one with no internal resistance; where the current supplied is only dependent on the external circuit. As the resistance of this circuit approaches zero, the ideal voltage source is able to supply a current approaching infinity according to ohm's law $V = IR$. Of course, this cannot ever be the case since then the source would be supplying an infinite amount of power $(I * V)$.

In reality, every voltage source has some internal impedance or resistance commonly referred to as output impedance. This impedance prevents the function generator from being able to break the laws of physics. Such non-ideal voltage sources can be modeled with some impedance in series as shown in figure 3.3. This becomes more important as an attached circuit draws an increasing amount of current. The voltage output of the function generator will drop as more and more current is drawn.

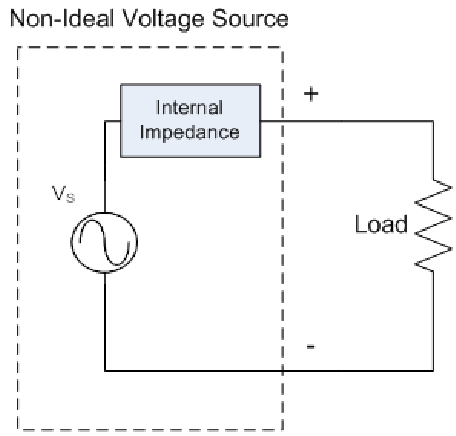

Figure 3.3: Non-Ideal Source

To accurately simulate a circuit, it is sometimes necessary to know the internal impedance of a source. In this lab calculating the internal resistance and reactance of the function generator is the next step.

1. Using the oscope, ensure that the signal generator is producing a 9V$_{pk-pk}$, 1kHz sine wave. The impedance of the oscope probe is very large, so the voltage observed is at what is called 'no load.' It is important to realize that the voltage will change when the circuit is loaded if nothing else is done. Since there is essentially no current flowing, there will be no voltage drop across the internal impedance and therefore the value you measure will be equal to V$_s$.
2. Record your measured value of V$_s$(Amplitude of Signal) ______________________________(3)
3. Now, using the breadboard, connect a 1kΩ load resistor to the function generator and measure the magnitude of the voltage across this resistor. Do not adjust the function generator in any way.
4. Record the voltage across the 1kΩ resistor using the oscope. ______________________________(4)
5. Now replace the 1kΩ resistor with a 220Ω load resistor.
6. Record the voltage across the 220Ω resistor. ______________________________(5)

Since the value of the load resistor and the voltage across it is known,the impedance can be solved for using a basic voltage divider equation.

$$|V_{load}| = \frac{R_{load}}{|R_{load}+Z_{out}|}|V_s|$$

Separating the internal impedance into resistance and reactance gives:

$$|V_{load}| = \frac{R_{load}}{|R_{load}+R_{out}+jX_{out}|}|V_s| = \frac{R_{load}}{\sqrt{(R_{load}+R_{out})^2+X_{out}^2}}|V_s|$$

In this simplified equation there are two unknowns, R_{out} and X_{out}, corresponding to the resistive and reactive parts of the internal impedance. With two unknowns another equation is needed to form a system of equations that can be solved simultaneously. The second equation can be found by using the measurements from the second resistor. Calling the 1kΩ resistor R1 and the 220Ω resistor R2 with the corresponding load voltages V1 and V2 the above equation can be adjusted to give both of the equations below.

$$\sqrt{(R_1 + R_{out})^2 + X_{out}^2} = R_1 \frac{|V_s|}{|V_1|}$$

$$\sqrt{(R_2 + R_{out})^2 + X_{out}^2} = R_2 \frac{|V_s|}{|V_2|}$$

Squaring both equations and then subtracting the second from the first, yields the following:

$$(R_1 + R_{out})^2 - (R_2 + R_{out})^2 = (R_1 \frac{|V_s|}{|V_1|})^2 - (R_2 \frac{|V_s|}{|V_2|})^2$$

1. This equation with only one unknown, R_{out}, can be solved. Using the measured values, solve for R_{out} and record it ____________________(6)
2. Using the calculated value for R_{out} solve for X_{out} using either of the above equations and record it ____________________(7)
   Note: X_{out} and R_{out} can be positive or negative. Assume positive.
3. Now record the total internal impedance of the function generator in both rectangular and polar form.
   Rectangular (Resistance and Reactance): __________ + j __________ Ω (8)
   Polar: __________ + $\angle$ __________ ^{o} (9)

3.5 Other Non-Ideal Components

Non-ideal aspects of inductors, capacitors, and resistors also come into play in circuit simulation and construction. Ideally, inductors and capacitors are made up entirely of reactance and do not dissipate any real power. In reality both components feature some parasitic qualities which include series resistances, capacitances and inductances for each element. These parasitics typically only become a problem at high frequencies currents.

One good example of this is the long windings of wire around the core of an inductor, which create a measurable resistance commonly referred to as equivalent series resistance (ESR). To see this, try using a digital multimeter to measure the resistance of a 10mH inductor.

Record the resistance here. ____________________(10)

The measured resistance is relatively small, but can certainly have an effect depending on the frequency of a circuit. To simulate a circuit completely, a model of an inductor with an ESR would be needed. This could be done by using a resistor in series with the inductor as shown in figure 3.4. However most simulation tools allow for the addition of DC resistance in the component properties for inductors (and capacitors).

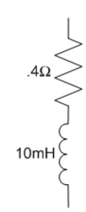

Figure 3.4: ESR Inductor

Similar to inductors, capacitors also exhibit some ESR although it is much harder to measure. A couple common ESR values for capacitors are in table 3.2. Non-ideal capacitor's can also be modeled using a small resistor in series with the capacitor.

Capacitance	Equivalent Series Resistance
$.1\mu F$	$.08\Omega$
$10\mu F$	$.03\Omega$

Table 3.2: Capacitor Resistance Values

3.6 Modeling Non-Ideality in LTSpice

As this lab has shown, there are no completely ideal electrical components. Every component has some parasitic properties that prevent it from behaving exactly as expected. In some cases, it can be important to accurately model these properties before constructing the circuit. LTSpice has mechanisms for accounting for all of the non-idealities in all of the basic components. To show this, let us create a non-ideal version of one of the circuits from lab 2, shown in Figure 3.5.

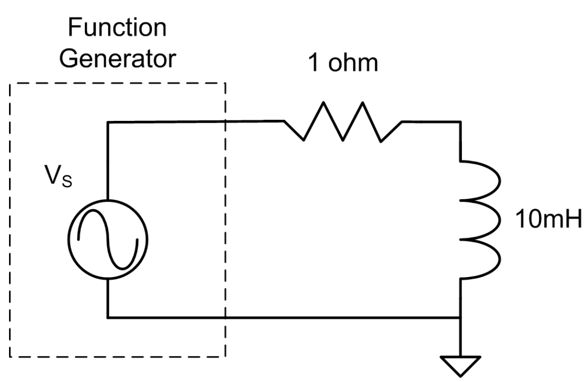

Figure 3.5: Non-Ideal Circuit

Begin by placing a 5V$_{Peak-Peak}$ 1kHz voltage source, 1Ω resistor and 10mH inductor on a new LTSpice circuit and connect them to match the above circuit. This is the ideal version of the circuit. Make a copy of the circuit on the same page. This will be used for the non-ideal version of this circuit. Now add parasitics to the circuit by adding components to simulate their presence.

First, right click on the voltage source to bring up the advance settings. On the right part of the window there is a section for parasitic properties of the source. Under the section titled "Series Resistance", fill in the output resistance of the function generator that was solved for earlier.

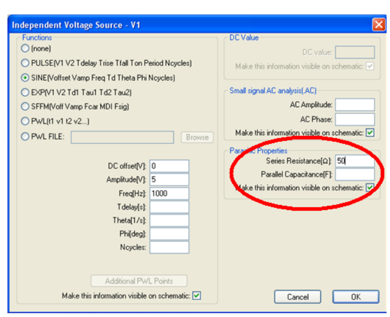

Figure 3.6: Using LTSpice for Non-Ideality

To be 100% accurate, the input reactance of the function generator should be identified as inductive or capacitive and converted to a parallel capacitance to be entered into LTSpice. For the purposes of this lab, the reactive portion of the input impedance will be ignored.

Next, right click on the non-ideal resistor to bring up the resistor settings. Under tolerance, enter 5 percent to model a real resistor.

Finally, right click on the non-ideal inductor and enter the value of ESR that was measured earlier.

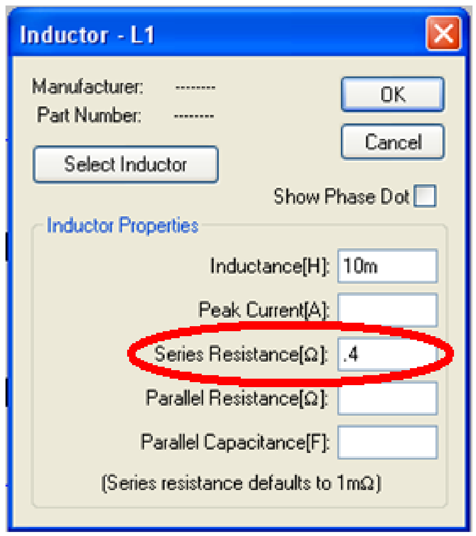

Figure 3.7: Series Resistance in LTSpice

Add a ground node to each circuit and then select Simulate → Run to bring up the simulation settings window. Enter 1.01 for the Stop Time and 1 for the Start Time and click OK to run the simulation.

Using the digital probes in LTSpice, view the waveforms of the current through the inductor on both the ideal and non-ideal circuit. There should be noticeable differences. Spend a few minutes looking at the differences between the two simulations in LTSpice. If the differences are not obvious, ask the course teaching assistant for help.

Print out Schematic and Simulation results to turn in with lab report
Make sure output graphs of each waveform is distinguishable with correct labels.

3.7 Study Questions

1. Calculate the equivalent series inductance of the function generator from the X_{out} determined earlier.
2. Calculate the peak instantaneous and RMS power dissipated through a 220Ω resistor supplied by a $9V_{peak-peak}$, 1Khz signal.
3. List a few examples of when it would be important to account for the non-ideal characteristics in your circuit design.

